CFD SIMULATION STUDY ON THE EFFECT OF WATER VELOCITY TOWARDS OIL LEAKAGE FROM SUBMARINE PIPELINES

AMARASINGAM A/L SELVARAJAH

Thesis submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Chemical Engineering (Gas Technology)

Faculty of Chemical & Natural Resources Engineering
UNIVERSITI MALAYSIA PAHANG

JULY 2014

ABSTRACT

This paper presents Computational fluid dynamic (CFD) studies on water velocity effect towards the time taken for migration of oil droplets to reach free surface. Computational Fluid Dynamic (CFD) simulations with FLUENT software 6.3.26 were simulated to detect the leakage process of oil spill from submarine pipeline to free surface. GAMBIT 2.4.6 mesh-generator is employed to perform all geometry generation and meshing. The velocity inlet of water (vs.) was varied whereas density of oil (kerosene liquid) was constant at 780kg/m^3 .A computational rectangular domain with length of 20m and height of 15m was simulated in Gambit 2.4.6. The mesh was generated and exported to Fluent. In the Fluent 6.3.26, the time taken for the oil droplets to reach free surface was observed by varying water inlet velocity; v_{w1}=0.02m/s, v_{w2}=0.04m/s,v_{w3}=0.08m/s respectively. Kerosene droplets reached free surface faster as the velocity of water inlet increased. Results were observed at 1000 number of time steps (iterations) with a step size of 0.1seconds. The leak size was shown to be 0.1meter, which was fixed at the beginning of the simulation conditions. **Justifications** were shown where oil droplets released from a greater leak width are easier to collision and have greater chance of gathering into large droplets, (Zhu et al., 2013). This is because at a larger face of leakage, the shear stresses increases, causing a larger displacement in oil migration. From the study, the dimensionless longest horizontal distance the kerosene droplets migrate when they reach the sea surface are analysed and the fitting formulas are obtained. With this, the maximum horizontal migration distance of oil at certain time is predicted, and a forecasting model is proposed in order to place the oil containment boom. This helps to detect the leakage more accurately and reduces cost of handling.

Key words: Computational fluid dynamic (CFD), computational domain, Gambit 2.4.6 and Fluent 6.3.26, Water velocity

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

<u>Greek</u>	
v_l	kinematic viscosity
µm	micrometre
m/s	Water velocity / oil leaking rate
m^3/s	Volume flux
wt %	moisture content
KJ/kg	Calorific Value
Kg/m3	oil density
Pa.s	Pascal .seconds
Diameters	(height, distance)
ρ	density
g	gravity

CFD	Computational Fluid Dynamics
RANS	(Reynolds-Averaged-Navier-Stokes) equations,
VOF	Volume of Fluid
FVM	(finite volume method)
MESH	2.4.6
FLUENT	6.3.26

1 INTRODUCTION

1.0 Brief introduction and problem statement

Oil had become one of the most important energy we have. Every day we will use hundreds of things that are made from oil. Therefore, the demand for this energy is quite large and increasing. This issue had led to the exploration of oil and then the construction of submarine pipeline system at the sea. The important issues related the submarine pipeline is the oil spill or leakage incident. These incidents usually present in pipelines due to several factors such as corrosion, flow erosion, or submarine landslide. This incident may lead to serious environmental issues especially to marine life and human health. As the largest accidental marine oil spills had occurred in Gulf of Mexico, around 4.9 million barrels of oil were released into the sea. Due to the months-long spill, along with adverse effects from the response and clean-up activities, extensive damage to marine and wildlife habitats, fishing and tourism industries, and human health problems have continued through 2014 (Tangley,2010).

Some research has forecasted the trajectory of submarine oil spill using radar galvanic current (Abascal et al., 2009), but the approach can only supply partial real-time information and may not support emergency behaviour for the influence of weather and night. Li and Yapa (2002), Øistein et al. (2003) and Dasanayaka and Yapa (2009) have also carried out the research on submarine oil ejecting, but they all aim at oil gas mixture and cannot contribute to forecasting oil spill greatly.

In this modern era, exploration is now extending into sensitive areas, in particular, offshore field. Washout and perforation failures are usually present in oil submarine pipelines due to corrosion or flow erosion. Then oil spills into marine environment from the leak, causing extensive damage to marine life, human health, and natural resources (Wang et al., 2013).

According to Xu and Wei (2013), oil spill accidents occurred at platform B and C of the Penglai 19-3 oilfield located in Bohai Sea It was estimated about 700 barrels of oil and 2500 barrels of mineral oil-based drilling mud were released at shallow water depths of 18m, causing a relatively high risk to the environment.

Once accidental oil leakages occur, a quick and adequate response in order to reduce the environmental consequences is required. (Biksey et al., 2010). Besides, laying oil containment boom, as a basic way to control oil dispersal, also depends on the rising velocity of oil droplets and the trend of spreading. Therefore, an exact prediction of oil spill process and dispersal could provide useful information for setting up oil containment boom and reducing the damage of future oil spills. (*Hongjun* et al., 2014)

An effective attempt has been made to observe the oil spill under the action of current and wave. However, the velocity of current in their study was uniform, which does not match with the actual shear velocity distribution under sea surface. And the actual hydrostatic pressure distribution was not used in their modelling. Moreover, the crucial parameter, the maximum horizontal migration distance of oil, was not considered in their research. (*Li* et al., 2013)

1.1 Motivation

The increasing oil spill accidents from submarine pipelines have caused severe damage to the aquatic life and health problems to mankind. A lot of research had been made manually and by simulation to study the effective measure in detecting a leakage. Because of the oil leakage from damaged submarine pipeline, the migration of oil flow along the depth direction is an important issue to address.

Hence, numerical simulation can provide detailed information on the hydrodynamics of oil flow, which is not easily obtained by physical experiments. CFD (computational fluid dynamic) model coupling with VOF (volume of fluid) method has been used to investigate the process of oil spill from submarine pipeline to free surface. The actual shear velocity distribution of current and the actual hydrostatic pressure distribution are considered in this study.

Detailed oil droplet and sea-surface in-formation could be obtained by the VOF model. By conducting a series of numerical simulations, effects of oil density, oil leaking rate, leak size and water velocity on the oil spill process are examined. Then, the dimensionless time required for oil droplets which have the longest horizontal migrate distance when they reach the sea surface and the dimensionless longest horizontal distance the droplets migrate when they reach the sea surface are analyzed and the fitting formulas are obtained.(Yadav et al.,2013 ; Arpino et al.,2009 ; Jalilinasrabady et al.,2013).

Summary

The topic was scoped from addressing the problem in the petroleum industry, way by identifying the problem of leakage in submarine pipelines.Then, an alternative solution using the CFD (computational fluid dynamic) model coupling with VOF (volume of fluid) method has been used to investigate the process of oil spill from submarine pipeline to free surface.

1.2 Objectives of Study

The main objective of this study is to investigate the time taken for oil droplets(s) to migrate along a horizontal distance up to free surface with varying water velocities (m/s) using the Gambit 2.4.6 and Fluent 6.3.26.

1.3 Scope of this research

The scopes of this study are to mainly study the effects of water velocity on the oil spill process. The method of study is by implementing computational fluid dynamics using the Gambit 2.4.6 and the Fluent Software. GAMBIT 2.4.6 mesh-generator is employed to perform all geometry generation and meshing. The whole computational domain is a rectangle with a length of 20 m and a height of 15 m. The length of computational domain is large enough, which is larger than the longest horizontal distance the oil droplets migrate when they reach the sea surface. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. The damaged submarine pipe with the outer diameter (D) of 0.6 m at both sides. . In the Fluent 6.3.26, the time taken for the oil droplets to reach free surface was observed by varying water inlet velocity; v_{w1}=0.02m/s, v_{w2}=0.04m/s,v_{w3}=0.08m/s respectively. Kerosene droplets reached free surface faster as the velocity of water inlet increased. Results were observed at 1000 number of time steps (iterations) with a step size of 0.1s.

1.4 Hypothesis

As the water velocity inlet (m/s) increases, the time taken for the oil droplets to reach free surface is much shorter.

1.5 Main contribution of this work

The following is the contribution:

- Contribution was prior to our supervisor's guidance in helping us learn and venture into the CFD simulation of Gambit and Fluent Software.
- And from this simulation study, I can be able to understand the factor of water velocity which affects the time for oil droplets to reach free surface.

1.6 Organisation of thesis

The structure of the reminder of the thesis is outlined as follow:

Chapter 2 provides an overview of oil leakage in pipelines underneath the ocean. A general description on the Computational Fluid Dynamics (CFD) and the Volume of Fluid (VOF) approach. This chapter also provides a brief review on previous study made on oil leakage myths. A comparison made on all the factors which directly affect the time period for the spills to reach the free surface.

Chapter 3 gives a review of the procedure involved in the simulation process. The mesh is generated and exported to the Fluent Software to simulate results. Results generated for varying velocities are compared to its standard case.

Chapter 4 gives a clear understanding of the effect of water velocity on the time taken for oil to reach the sea-surface while leaving other parameters of oil density, oil leak rate and leak size constant. It is attributed to the increasing kinetic velocity of oil droplets. Hence, the larger the water velocity, the more obvious the trajectory of oil flow skewed to the downstream. The reason is that high-speed water exerts more shear stress on oil droplets and transfers more kinetic energy to oil droplets (kerosene liquid).

Chapter 5 draws together a summary of the thesis and outlines the future work which might be derived from the model developed in this work.

2 LITERATURE REVIEW

2.0 Screening Route

My literature study was on the oil leakage in submarine pipelines which endanger the environment and aquatic life. In addition, research was also completed on Computational Fluid Dynamics (CFD). The study conducted did not neglect the environmental aspects and economy imprecision. Finally, the literature centered on the effect of water velocity towards oil migration in a horizontal distance to the free surface.

2.1 Oil Leakage

Over the past few decades, several major U.S. oil spills have had lasting repercussions that transcended the local environmental and economic effects. The April 2010 oil spill in the Gulf of Mexico has intensified interest in many oil spill-related issues. Prior to the 2010 Gulf spill, the most notable example was the 1989 *Exxon Valdez* spill, which released approximately 11 million gallons (260,000 barrels) of crude oil into Prince William Sound, Alaska. The *Exxon Valdez* spill produced extensive consequences beyond Alaska. According to the National Academies of Science, the *Exxon Valdez* disaster caused “fundamental changes in the way the U.S. public thought about oil, the oil industry, and the transport of petroleum products by tankers ‘big oil’ was suddenly seen as a necessary evil, something to be feared and mistrusted; Jonathan L (2012)

Offshore production constitutes a major portion of the overall oil and gas production. Offshore oil and gas production is more challenging than land-based installations due to the remote and harsher environment. Other than the production challenges, environmental risks due to oil spills pose major challenges. An “oil spill” usually refers to an event that led to a release of liquid petroleum hydrocarbon into the environment due to human activity and is a form of pollution. Oil spills usually include releases of crude oil from tankers, offshore platforms, drilling rigs and wells, as well as spills of refined petroleum products (such as gasoline, diesel) and their by-products, and heavier fuels used by large ships such as bunker fuel, or the spill of any oily white substance refuse or waste oil. Spills may take months or even years to clean up. (*Agrawal* et al., *2011*)

There are several factors that may cause the oil spill from submarine pipeline.

Table 2-1: The **factors** of how oil spill can occur from damaged submarine pipeline

Factor	Explanation	Author
Submarine landslide	This is happen due to high of sedimentation rates and usually occurs on steeper slopes. This landslide can be triggered by earthquakes in the sea. When the soil around the piping system is subjected to a slide, and give the result of displacement at high angle to the pipeline, the pipe will severe bending. This will cause tensile failure.	Palmer & King (2008)
Ice issues	This happen to submarine pipeline system in low temperature water especially in freezing waters. In this case, the floating ice features often drift into shallower water. Therefore their keel comes into contact with the seabed. When this condition happen, they will scoop the seabed and came hit the pipeline	Croasdale K. (2013)
	Stamukhi can also damage the submarine pipeline system. Stamukhi is a grounded accumulation of sea ice rubble that typically develops along the boundary between fast ice and the drifting pack ice. This stamukhi will exert high local stresses on the pipeline system to inducing the excessive bending.	Croasdale K. (2013)
Ship anchors	Ship anchors are a potential threat to submarine pipelines, especially near harbours. This anchor will give high damage to the pipeline due to their massive weight.	
Corrosion	For small size lines, additionally, failures due to external corrosion were more frequent compare than internal corrosion. However in medium and large-size lines, failures due to internal corrosion were more frequent than those due to external corrosion.	J. S. Mandke (1990)

Table 2-2: Oil Spills of 100,000 Tons (640,000 Barrels), or More
(*International Tanker Owners Federation, 2001 New York Times Almanac*)

Date	**Cause**	**Location**	**Barrels Spilled**	**Rank, by spilled volume**
1942	German U-boats attacks on tankers after U.S enters World War 11	U.S. East Coast	590,000	4
1970	Tanker *Othello* collides with another ship	Tralhavet Bay, Sweden	60,000 to 100,000	15
1994	Pipeline bursts, oil enters rivers that flow into Arctic Ocean	Near Usinik, Russia	312,500	5

2.2 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), software tools (solvers, pre- and post-processing utilities), and numerical methods (discretization and solution techniques). (*Wesseling* et al., 2001)

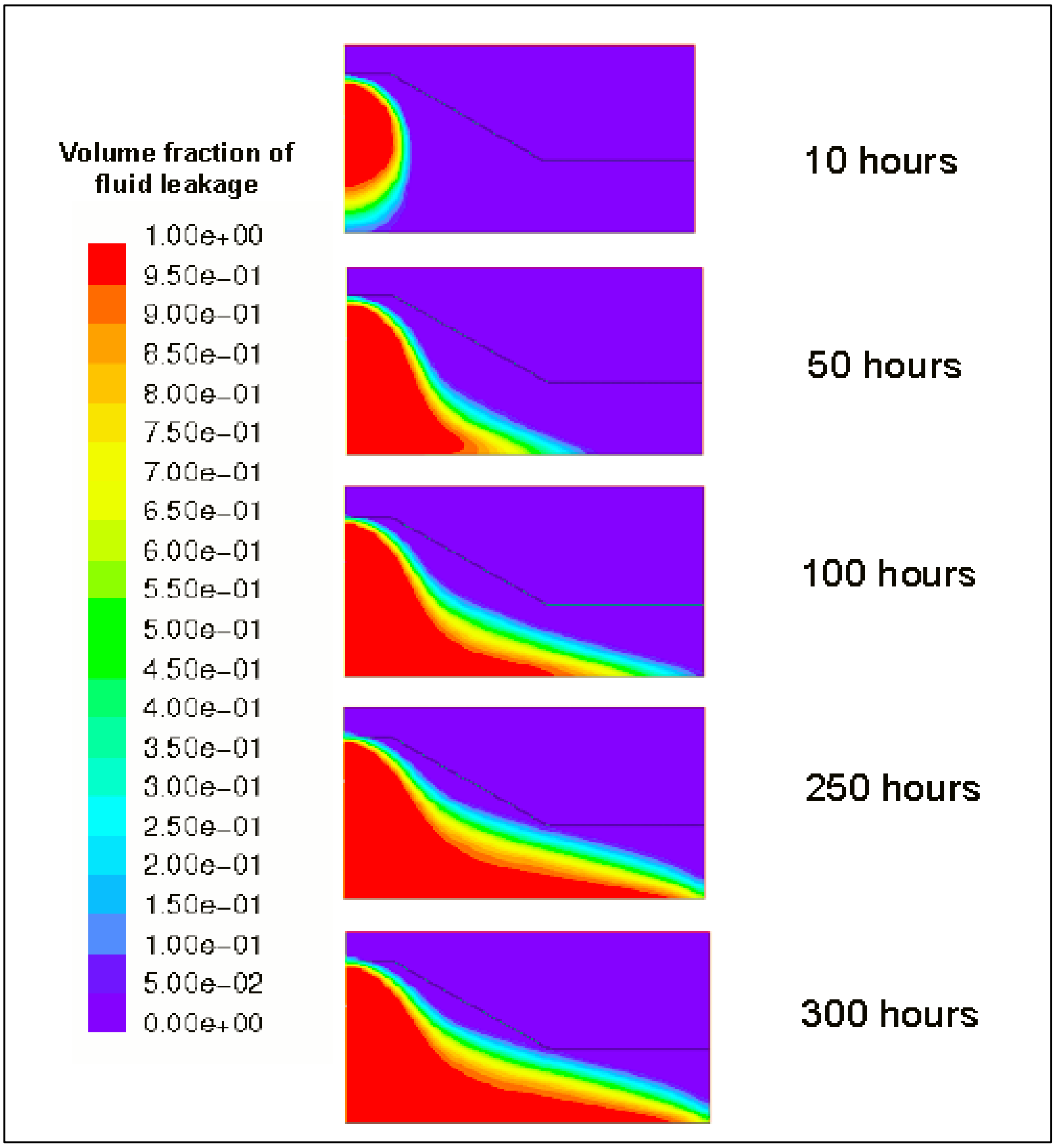
2.2.1 Oil Spillage

By definitions, Leakage is the accidental admission or escape of liquid or gas through a hole or crack from the pipeline. Spillage is an accidental release of oil /liquid petroleum hydrocarbon to the sea from tankers at larger amounts than a leak, which may end in Pollution. Both definitions were derived from the *Oxford Dictionary 2010*.Alexander, C. (2005) through Stress Engineering Service (SES) performed a more rigorous investigation to determine what conditions were required to produce daylighting, the significance of which involved quantifying the estimates of leak duration and the petroleum volumes at Houston, Texas. This effort integrated assumptions and data from prior analyses to assess the effects of time-dependency using computational fluid dynamics (CFD) modeling techniques.

Table 2-3: CFD Modeling Scenarios; Alexander, C. (2005)

Scenario	Description	Volume of product released	Daylight (YES or NO)
1	Continuous leaking (330 hours)	24685 gallons 240 ft^3/day-330 hours	YES
2	Leak rates based on tabulated existing data considering two different pressure levels a. Pipeline beginning pressure (***667 hours***) b. Static head end pressure (***667 hours***)	4,940 gallons (a) 6,212 gallons (b)	NO (a) NO(b)

Table 2-4: CFD contour plots with continuous leaking at 240 ft^3/dayAlexander, C. (2005)



According to another research from Shehadeh (2012), the research focused on studying the velocity magnitude (V), total pressure (P), and turbulence intensity (I) for the hole of 5 mm diameter in a pipe of 3.5 mm wall thickness at ambient temperature. The study is to put forward the relationship between these parameters and the leakage mass flow rate (m_{leak}).A new CFD model was applied to simulate leakage in water pipeline by means of turbulence.

All zones are solved by initializing the entire flow field using the k-epsilon realizable model, as shown in Figure2-1.

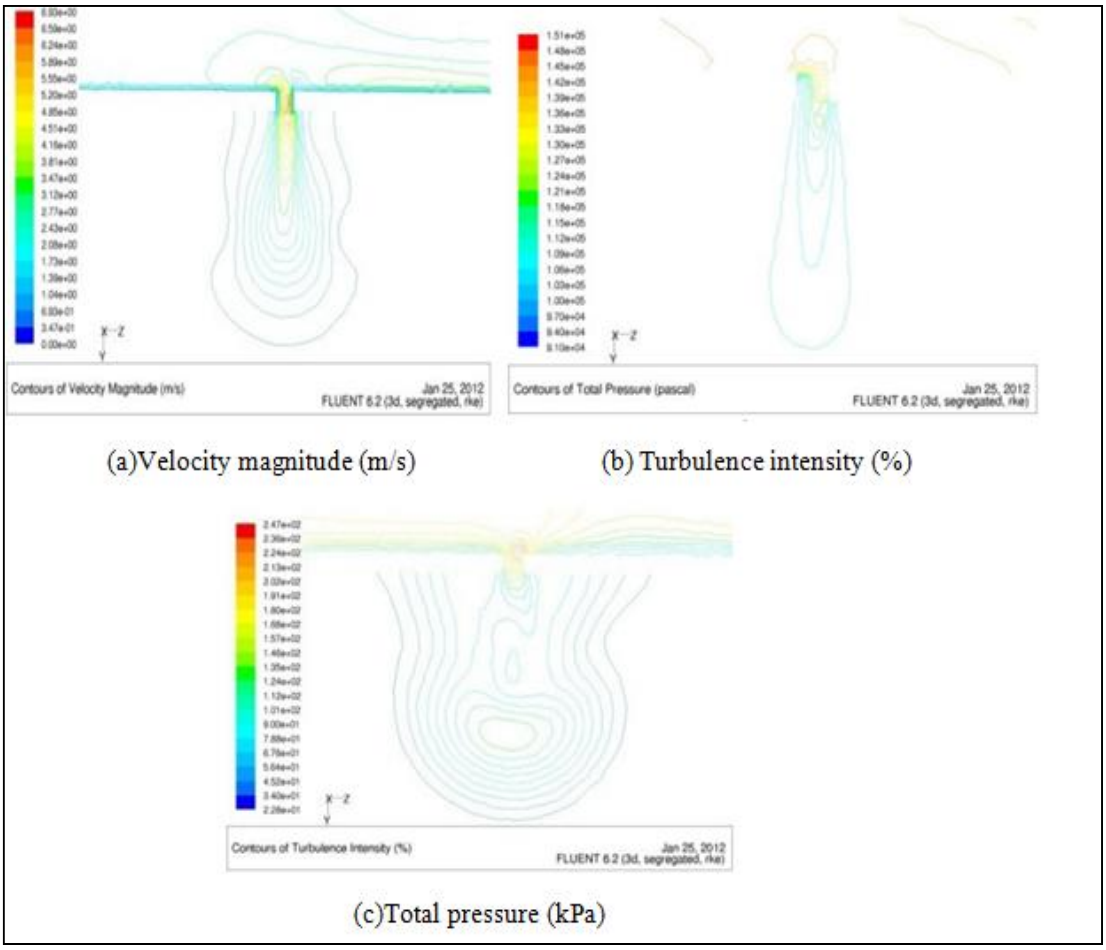


Figure 2-1: Fluent Simulation using k-epsilon turbulence model. (Shehadeh *et al.*, 2012)

Table 2-5 below depicts on the parameters used to run the fluent software in detecting water leakage.

Table 2-5: Parameters used in the Fluent Simulation (Shehadeh*et al.*, 2012).

Scenario	V_{in} (m/s)	P_{out}(kPa)	M_{in} kg/s	M_{leak} in kg/s	M_{out} in kg/s	$V_{m/s}$		P(kPa)	I(%)
1	2.7	100	5.96	0.1	5.86	Min	0	91.3	22.8
						Max	6.93	150.92	246.8
2	3.2	130	7.13	0.17	6.96	Min	0	81.62	26.9
						Max	11.23	249.01	369.7
3	3.7	155	8.24	0.22	8.02	Min	0	75.12	31.3

						Max	14.2	340.09	462.0

Hence, the turbulence intensity has a great effect on monitoring of pipeline, for instance leakage using novel technique such as acoustic emission [15] and ultrasonic techniques [16].The study has predicted correlations between the mass flow rate of the leakage and the various parameters of the pipeline system.

2.2.2 Experiments versus Simulations

Table 2-6: Comparison of experimental and simulation runs (*Wesselling et al., 2001*)

Experiments	**Simulations**
Quantitative description of flow phenomena using measurements • For one quantity at a time • At a limited number of points and time instants • For a laboratory-scale model • For a limited range of problems and operating conditions Error sources: measurement errors, flow disturbances by the probes	Quantitative prediction of flow phenomena using CFD software • For all desired quantities • With high resolution in space and time • For the actual flow domain • For virtually any problem and realistic operating conditions Error sources: modeling, discretization, iteration, implementation

2.2.3 The Finite Volume Method

A method for discretizing the transport equations commonly implemented in CFD codes are the finite volume method (FVM). In a FVM, the computational domain is divided in control volumes and conservation principles are applied to each control volume. This ensures conservation, both in each cell and globally in the domain, which is a great advantage of the FVM? Using FVM also allows for the use of unstructured grids which decreases the computational time. (*Stenmark et al., 2013)*

2.3 Multiphase Flow Theory

Multiphase flow is flow with simultaneous presence of different phases, where phase refers to solid, liquid or vapor state of matter. There are four main categories of multiphase flows; gas-liquid, gas-solid, liquid-solid and three-phase flows.
(*Thome, (2004))*

2.3.1 VOF Model Approach

A third modeling approach is the volume of fluid (VOF) method. VOF belongs to the Euler-Euler framework where all phases are treated as continuous, but in contrary to the previous presented models the VOF model does not allow the phases to be interpenetrating. The VOF method uses a phase indicator function, sometimes also called a colour function, to track the interface between two or more phases. The indicator function has value one or zero when a control volume is entirely filled with one of the phases and a value between one and zero if an interface is present in the control volume. Hence, the phase indicator function has the properties of volume fraction. The transport equations are solved for mixture properties without slip velocity, meaning that all field variables are assumed to be shared between the phases. To track the interface, an advection equation for the indicator function is solved. In order to obtain a sharp interface the discretization of the indicator function equation is crucial. (*Stenmark et al., 2013*)

2.4 Software

For geometry and mesh generation the ANSYS software ICEM CFD was used.

2.4.1 ICEM CFD

ICEM CFD is meshing software. It allows for the use of CAD geometries or to build the geometry using a number of geometry tools. In ICEM CFD a block-structured meshing approach is employed, allowing for hexahedral meshes also in rather complex geometries. Both structured and unstructured meshes can be created using ICEM CFD. (*Stenmark et al., 2013*)

2.4.2 Fluent

The Fluent solver is based on the centre node FVM discretization technique and offers both segregated and coupled solution methods. Three Euler-Euler multiphase models are available; the Eulerian model, the mixture model and the VOF model. In addition, one particle tracking model is available.

As mentioned in Section 2.3.1, the discretization of the volume fraction equation is crucial in a VOF method to keep the interface sharp. The choice of discretization method can have a great influence on the results in other multiphase models as well. To resolve this issue, Fluent has a number of discretization techniques implemented specifically for the volume fraction equation. Several methods are also available for spatial discretization of the other transport equations.

To model interphase transfer there is both a number of drag models available along with other transfer mechanisms such as lift forces and turbulent dispersion Fluent offers three main approaches to model dispersed phases with a two-fluid formulation. With the default settings it is assumed that the dispersed phase has a constant diameter or a diameter defined by a user-defined function. With this setting, phenomena such as coalescence and breakage are not considered. (*Stenmark et al., 2013*)

2.5 Computational Domain and Mesh

A sketch of the geometry (a) and numerical grid for computational domain (b) investigated in this study. GAMBIT 2.4 mesh-generator is employed to perform all geometry generation and meshing. The whole computational domain is a rectangle with a length of 20 m and a height of 15 m. The length of computational domain is large enough, which is larger than the longest horizontal distance the oil droplets migrate when they reach the sea surface. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. In the computational domain, the damaged submarine pipe with the outer diameter (D) of 0.6 m on both sides, the most common diameter of submarine pipe used in Bohai oilfield, is located in the sea bed. The speed of the water velocity (m/s) is a variable ranging from 0.01 m/s to 0.09 m/s, in order to examine the effect of water velocity. (*Zhu et al., 2013*)

Table 2-7:Simulation cases, in which variables of oil density, oil leaking rate, diameter of leak, and maximum water velocity are varied (*Zhu et al., 2013*)

Case	Oil density(kg /m^3)	The maximum water velocity(m/s)	Oil leaking rate(m/s)	Diameter of leak(m)	Volume flux of leaking oil(m^3/s)	Flux multiple(comparing with case 12)
1	780	0.1	2	0.05	0.003925	25
2	810	0.1	2	0.05	0.003925	25
3	840	0.1	2	0.05	0.003925	25
4	870	0.1	2	0.05	0.003925	25
5	900	0.1	2	0.05	0.003925	25
6	930	0.1	2	0.05	0.003925	25
7	960	0.1	2	0.05	0.003925	25
8	870	0.1	1	0.05	0.0019625	12.5
9	870	0.1	3	0.05	0.0058875	37.5
10	870	0.1	4	0.05	0.00785	50
11	870	0.1	5	0.01	0.0098125	62.5
12	870	0.1	2	0.02	0.000157	1
13	870	0.1	2	0.03	0.000628	4
14	870	0.1	2	0.04	0.001413	9
15	870	0.1	2	0.05	0.002512	16
16	870	0.04	2	0.05	0.003925	25
17	870	0.07	2	0.05	0.003925	25

2.6 Effect of oil density on Oil spills

As the density of oil and seawater is almost the same, when current velocity rises continuously with the same operating pressure, the influence of sea current is strengthened. Meanwhile, the influence of buoyancy is relatively weakened. When the current velocity is low (u = 0.1m/s), as shown in Fig. 2-2, the influence of buoyancy becomes dominant to the rising spilled oil. When the current velocity is u = 0.3, u = 0.5 and u = 0.8 m/s, respectively as shown in Fig. 2-3 to 2-5, the influence of sea current dominates obviously and oil particles move with sea current after spilled immediately. Therefore, the higher current velocity is, the longer submarine drift distance is with its respective densities. (Li *et al.*, 2013)

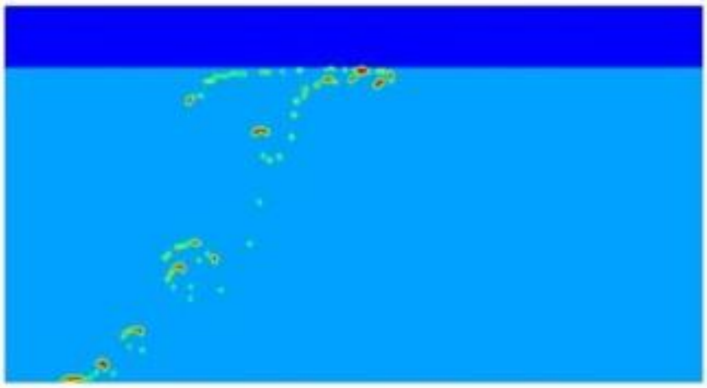


Figure 2-2: Distribution of oil-water-gas (t = 56s, u = 0.1m/s, P = 101000pa) (Li *et al.*, 2013)

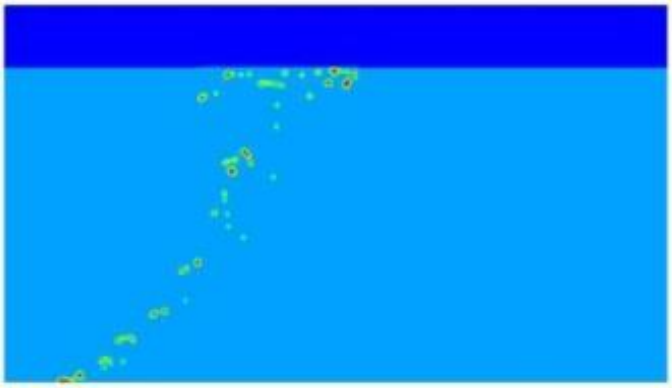


Figure 2-3: Distribution of oil-water-gas (t = 60s, u = 0.1m/s, P = 100800pa) (Li *et al.*, 2013)

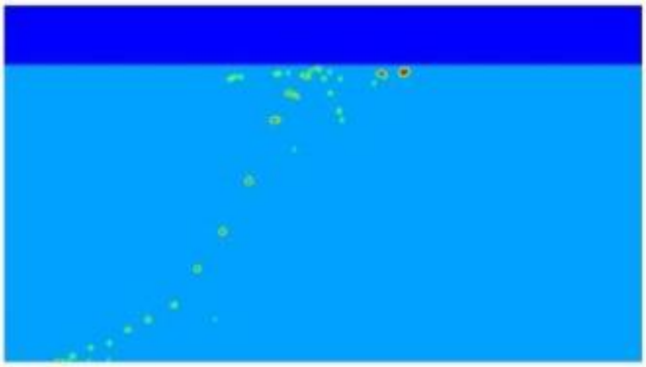


Figure 2-4: Distribution of oil-water-gas (t = 80s, u = 0.1m/s, P = 100600pa) (Li *et al.*, 2013)

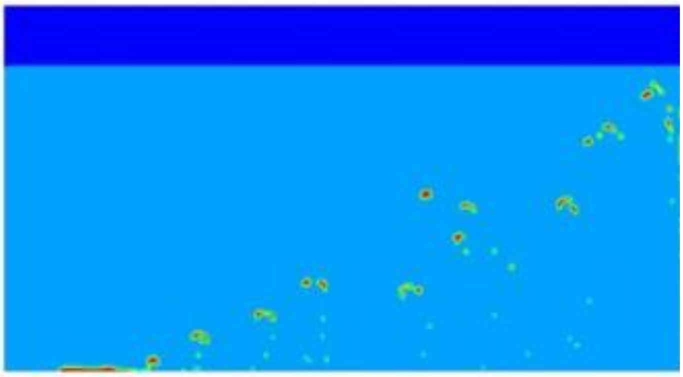


Figure 2-5: Distribution of oil-water-gas (t = 80s, u = 0.3m/s, P = 101000pa) (Li *et al.*, 2013)

3 MATERIALS AND METHODS

3.1 Overview

This paper is about to study the oil flows from damaged submarine pipelines with different water velocities. First and foremost, CFD (computational fluid dynamic) model coupling with VOF (volume of fluid) method has been used to investigate the process of oil spill from submarine pipeline to free surface. The actual shear velocity distribution of current and the actual hydrostatic pressure distribution are considered in this study. Detailed oil droplet and sea-surface in-formation could be obtained by the VOF model. Effects of oil density, oil leaking rate, leak size and water velocity on the oil spill process are examined.

3.2 Simulation Methodology

3.2.1 Governing equations

The VOF approach is based on the solution of one momentum equation for the mixture of the phases, and one equation for the volume fraction of fluid. In this study, volume of fluid functions F_w and F_o are introduced to define the water region and the oil region, respectively. The physical meaning of the F function is the fractional volume of a cell occupied by the liquid phase. For example, a unit vale of F_w corresponds to a cell full of water, while a zero value indicates that the cell contains no water. The fraction functions F_w and F_o are described as follows:

$$\boldsymbol{F_w}=\frac{\boldsymbol{V_w}}{\boldsymbol{V_c}} \qquad \textbf{Equation 3.2(a)}$$

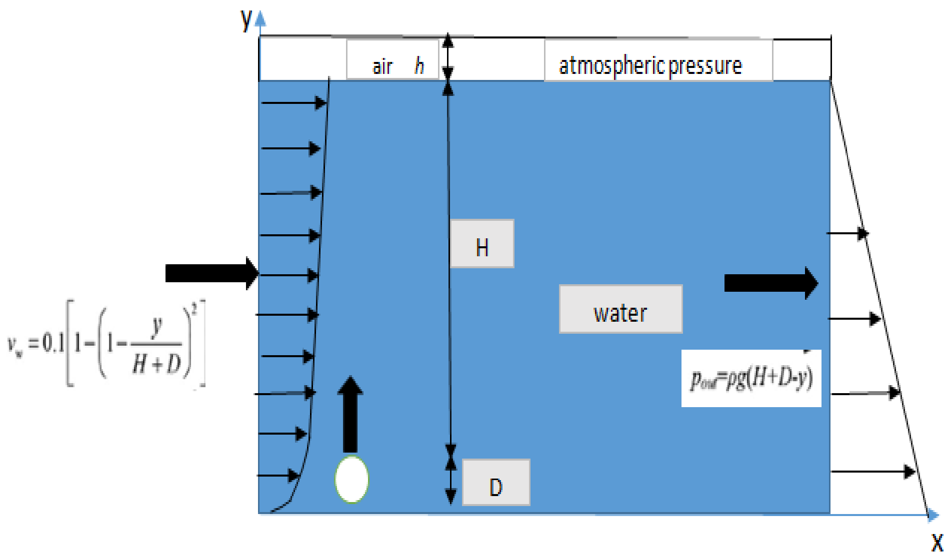
$$\boldsymbol{F_o}=\frac{\boldsymbol{V_o}}{\boldsymbol{V_c}} \qquad \textbf{Equation 3.2(b)}$$

where F_o and F_w are oil and water fractional function, respectively, V_c, V_o and V_w represent volume of a cell, volume of oil inside the cell and volume of water inside the cell, respectively**.**

3.2.2 Computational Domain and Mesh

(a) Gambit 2.4

A two-dimensional flow simulation is accurate enough to capture the maximum horizontal migration distance. Using Gambit 2.4, a rectangular computational domain was created with a length of 20 m and height of 15 m. Water occupies the lower region with height of 14.5 m, while air occupies the upper region. In the computational domain, the damaged submarine pipe with the outer diameter (D) of 0.6 m was displayed and transformed to a coordinate of (-9,-7, 0). There is a leakage hole on the top of pipe, opening upwards. The size of the leakage hole (d) is fixed at 0.1m. Then, a paved quadratic mesh with 0.5 spacing was generated for the leak and domain at one time. Progressive mesh is used to capture the near-leak flow properties. A suitable grid density is reached by repeating computations until a satisfactory independent grid is found. The mesh is then exported to be used to generate results in the Fluent 6.3.26. Figure 3-1 below shows (a) sketch of the geometry and (b) numerical grid for computational domain investigated in this study.



(a)